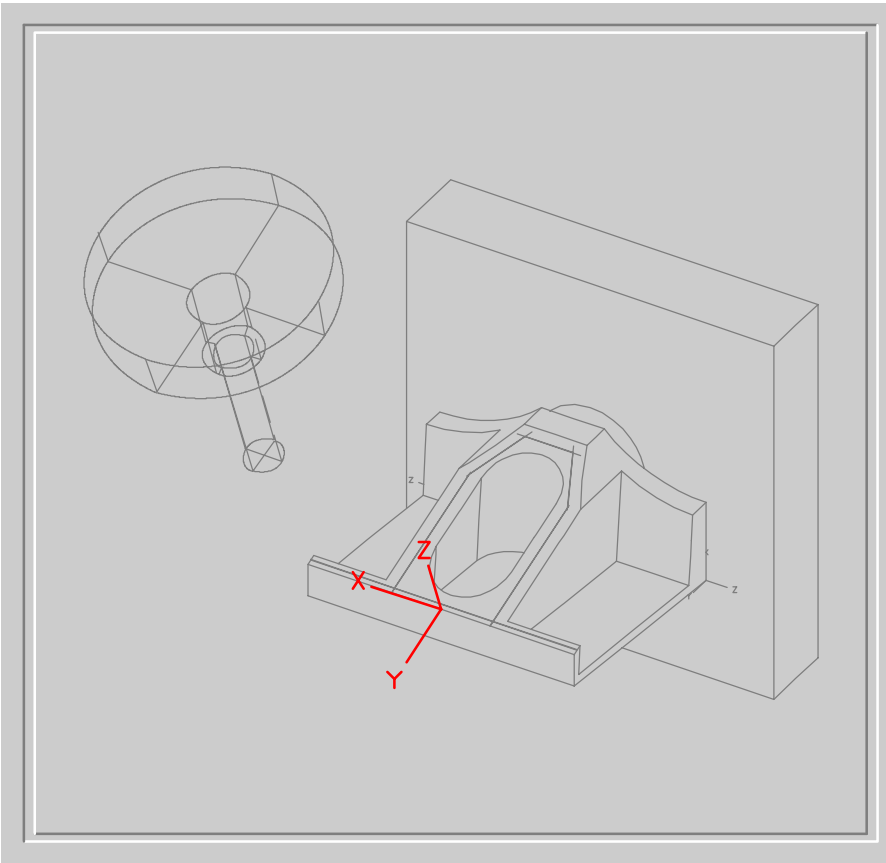


Indexing

I-DEAS® Tutorials: Milling Projects and Turning Projects

In previous tutorials, you learned how to machine a part using the global–space machining coordinate system as the origin. This tutorial teaches you how to add a 2-, 2 1/2-, and 3-axis machine tool to your setup. You'll also learn how to specify 5-axis positioning.

Do this tutorial if you have a machine tool with more than three axes. If you have three axes or fewer, do the tutorial called Using Multiple Setups.

Learn how to:

- define a machine
- create operation coordinate systems
- pick coordinate systems for operations

Before you begin...

Prerequisite tutorials:

- all tutorials under the Modeling Fundamentals menu
- Introduction to Generative Machining
- Building a Setup Assembly
- Generating In-process Stock and Checking Validity
- Working with Tools and Tool Catalogs
- Picking Holes
- Setting Machining Parameters for Hole-making Operations
- Creating Face Mill and Volume Clear Operations
- Creating Manual Milling Operations

The file you need for this tutorial is distributed with the product. You must copy it into your local directory.

Move to the local directory where you want to copy the file. Then:

In UNIX:

```
cp $SDRC_INSTL/examples/nc/tut_index.arc .
```

In Windows:

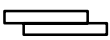
```
copy %SDRC_INSTL%\examples\nc\ tut_index.arc .
```

If you can’t copy the file, you may have to set up the variable needed to copy from the I−DEAS installation.

```
. sdrc_oadev
```

If you can’t access the file, contact your system administrator. The file may not be installed.

If you did not start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.



File
Open

Model File Name: any unique name

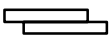
OK

Make sure you're in the following application and task:

Application: Manufacturing

Task: Generative Machining

Set your units to inches.



Options
Units

Inch (pound f)

Import the archive file that contains the parts and tools that you need to complete this tutorial. Importing an archive file can take several minutes. Be patient.

File
Import

Import Selections form

I–DEAS Archive File

OK

File Name Input form

Archive file name: tut_index.arc

OK

The Manufacturing application quits, an informational message is displayed (the message will dismiss automatically), and the archive file is imported.

Import Archive File Status

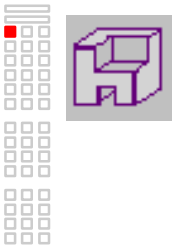
Check *I-DEAS List*.

Be sure to check the List region to be sure that the parts imported properly.

Dismiss

A second informational message is displayed (the message will dismiss automatically) and the Manufacturing application starts.

Create a job.



NC Job Create form

Job Name: Index Machine Bracket

OK

Add the part to the job.

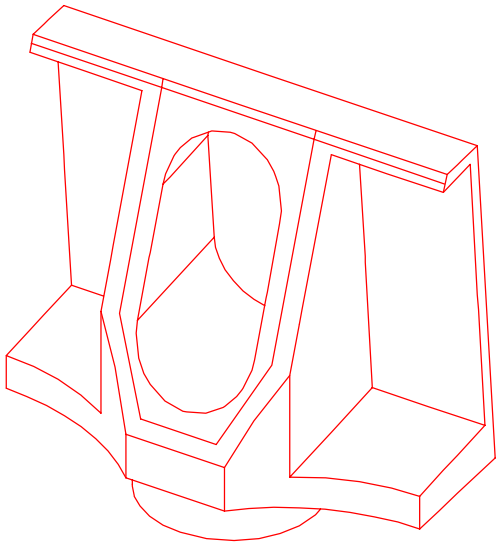
Get

From Bin/Library

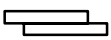
Select Part/Assembly form

tut_bracket_with_wireframe

OK



Recovery Point



File
Save

Warning!

If you're prompted by I-DEAS to save your model file, respond:

No

Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

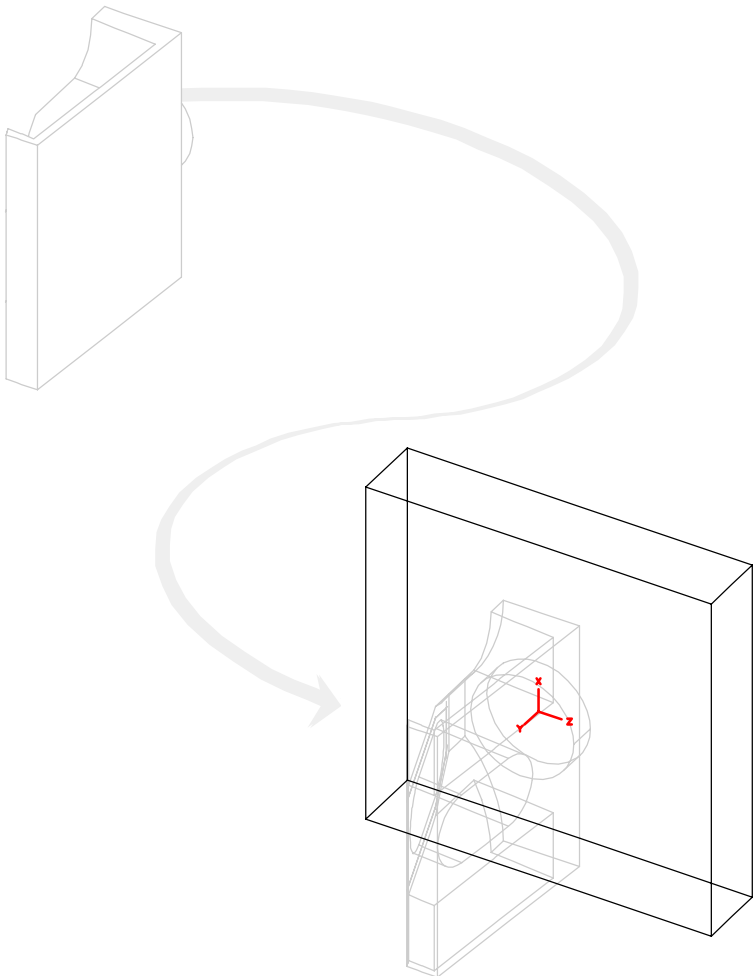
Hint

To reopen your model file to the previous save, press Control-Z.

Define a machine

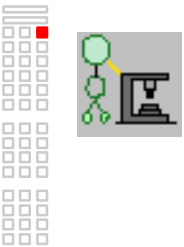
In the next steps, you'll add a machine instance to your setup and define its origin, orientation, and number of axes. Then you'll add a coordinate system to the machine. By adding a coordinate system, you make it capable of 2, 2 1/2, or 3-axis machining. You'll also modify the settings for the machine to allow 4- and 5-axis positioning.

For brevity, the machine has already been positioned relative to the part for you.



What: Add a machine to the setup assembly.

How:



Add Machine form

Get Machine From Bin/Library

Select Part/Assembly form

tut_base

OK

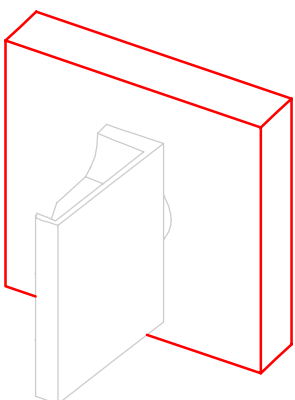
I-DEAS Warning

Dismiss

Name form

Name: Machine Assembly

OK



Don’t close the Machine Data: Mill, General form.

What: Create a machine coordinate system to specify the origin. The Z axis represents the axis of rotation for the tool spindle.

How:

Machine Data: Mill, General form

Type: Mill

Create MCS

Pick anywhere on the machine instance.

Origin

Between

1 *V1*

2 *V6*

X Axis

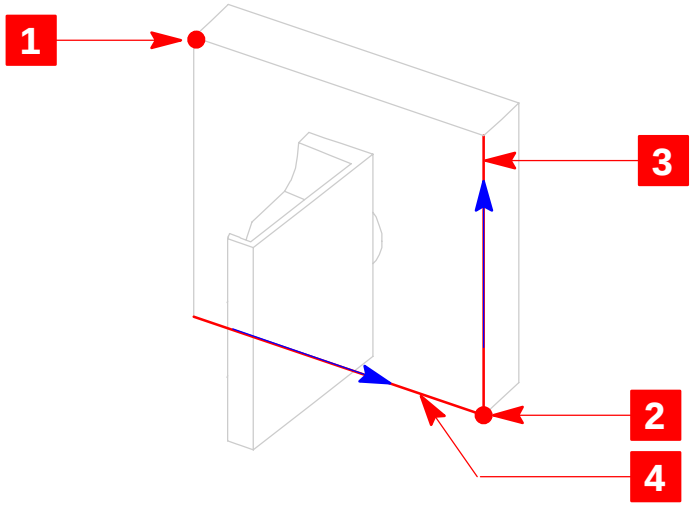
3 *E9*

Arrow direction: *Yes* or *No*.

Z Axis

4 *E7*

Arrow direction: *Yes* or *No*.



Don’t close the Machine Data: Mill, General form.

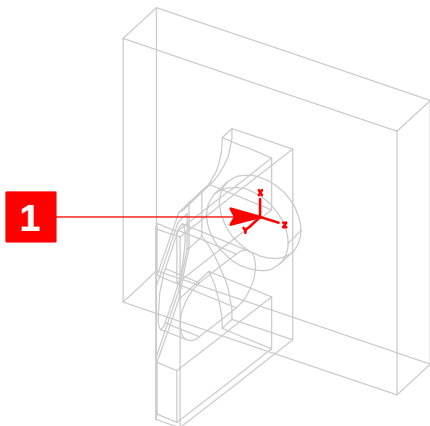
What: Pick the coordinate system to define it as the origin, or program zero.

How:

Machine Data: Mill, General form

Pick MCS

1 origin of coordinate system



General
Kinematics

4th Axis—On

Head
Table

5th Axis—On

Head
Table

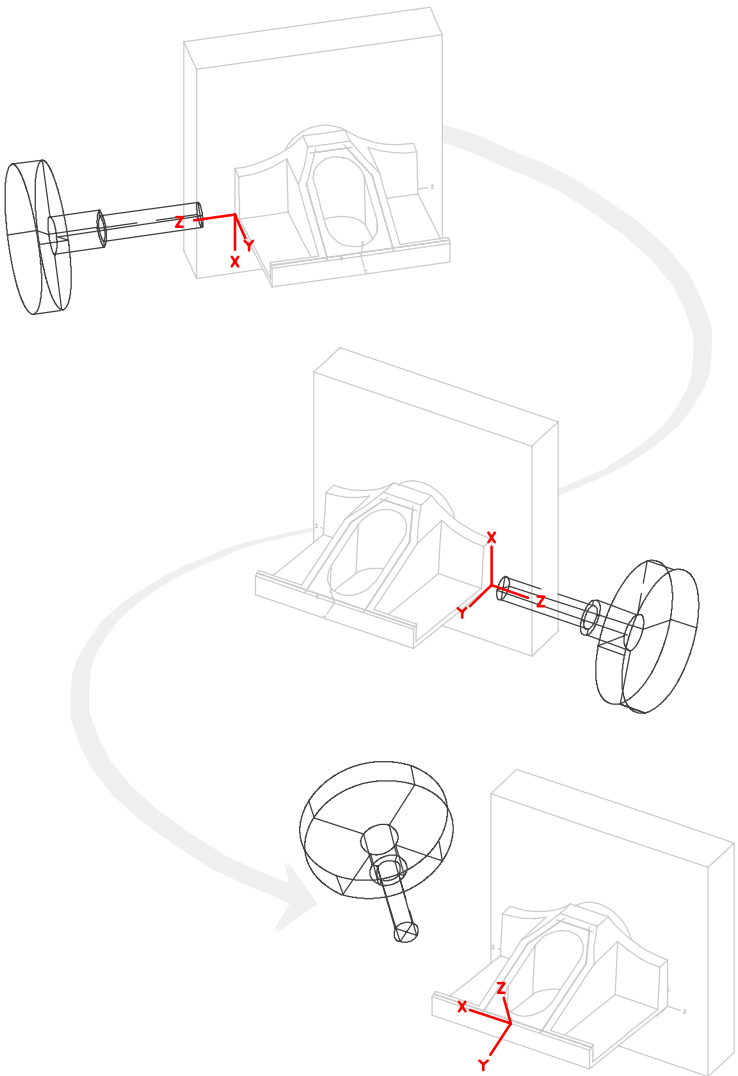
OK

Recovery Point

File
Save

After you've set 4- or 5-axis positioning, you must specify the positions to where the machine will rotate. You define these rotations by adding operation coordinate systems relative to the surfaces that are to be machined. The orientation of the Z axis represents the position of the tool spindle at each rotation.

In the next steps, you'll create three operation coordinate systems that represent three rotations of the machine tool.



What: Create an operation coordinate system for the first rotation. This coordinate system will be used to machine the pocket nearest to it.

How:

1 anywhere on the part

Origin

2 *V4*

X Axis

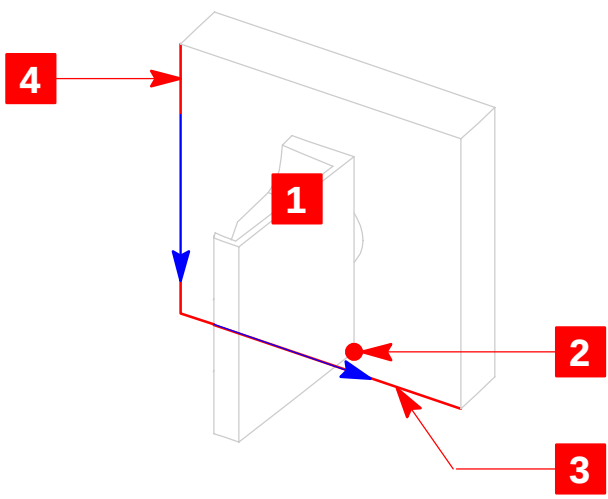
3 *E7*

Arrow direction: *Yes* or *No*.

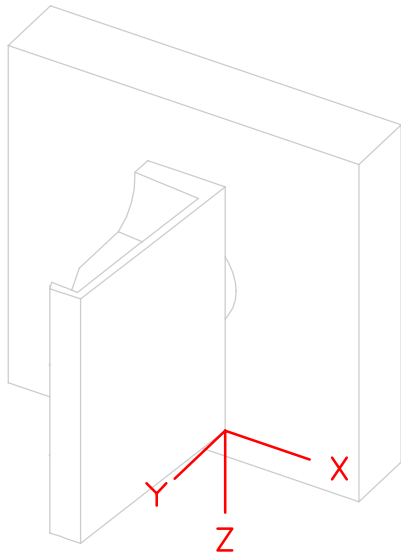
Z Axis

4 *E11*

Arrow direction: *Yes* or *No*.

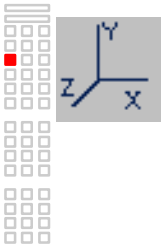


Result



What: Create a second operation coordinate system. This coordinate system will also be used to machine a pocket.

How:



1 anywhere on the part

Origin

2 *V3*

X Axis

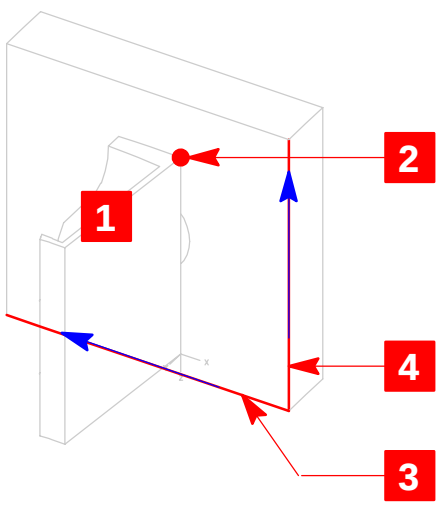
3 *E7*

Arrow direction: *Yes* or *No*.

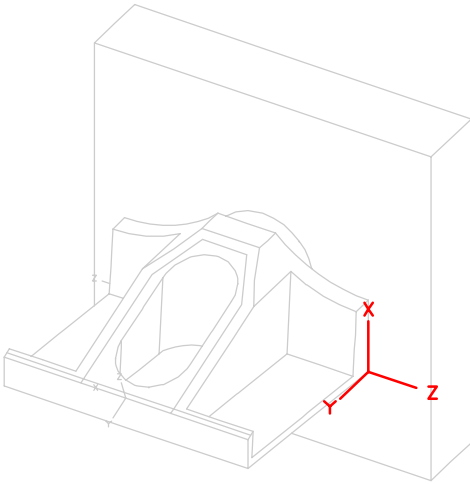
Z Axis

4 *E9*

Arrow direction: *Yes* or *No*.

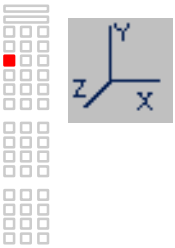


Result



What: Create a third operation coordinate system.

How:



1 anywhere on the part

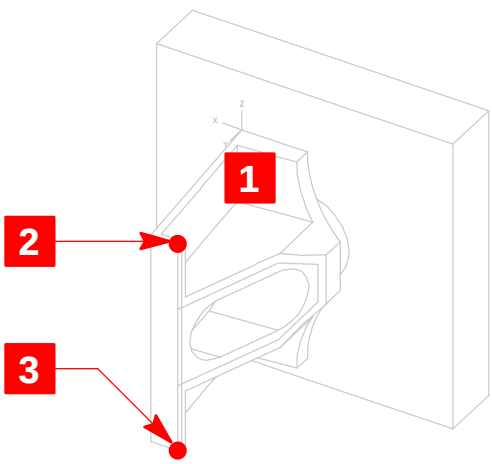
Origin

Between

2 V17

3 V15

Continue creating the coordinate system on the next page.



What: Continue creating the coordinate system.

How:

X Axis

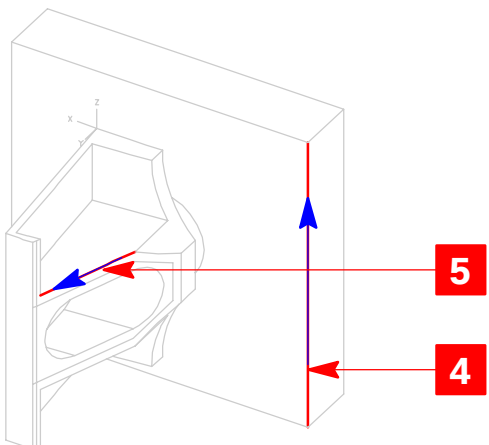
4 *E11*

Arrow direction: *Yes* or *No*.

Y Axis

5 E38

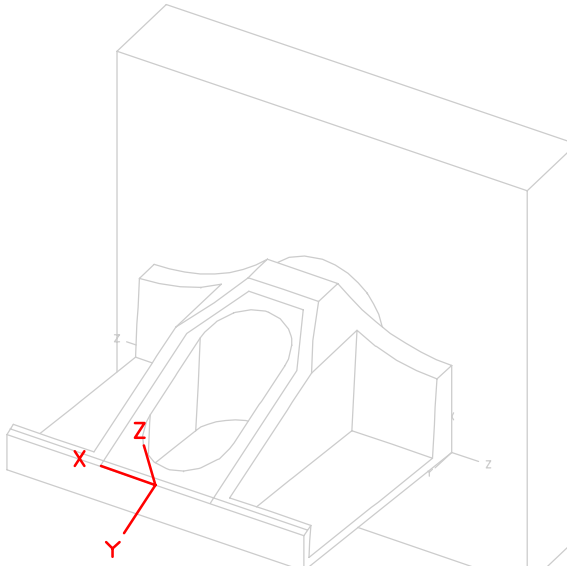
Arrow direction: *Yes* or *No*.



Recovery Point

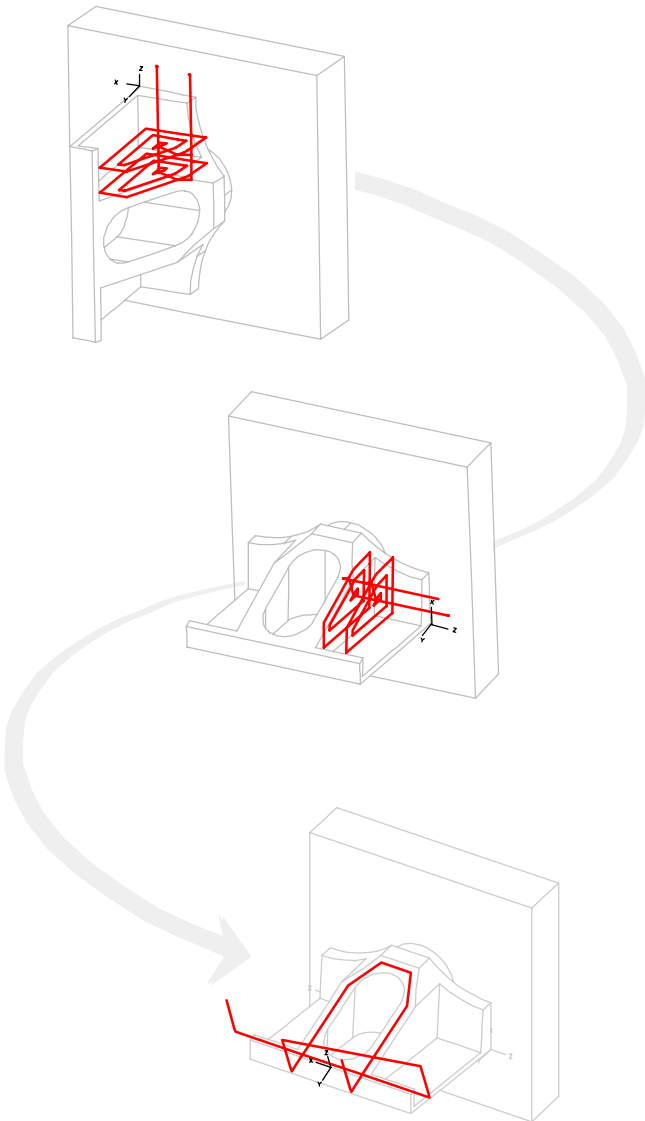
File
Save

Result



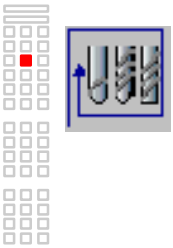
In the next steps, you'll create three operations, each with a separate index position. To reach the selected surfaces, you'll pick a different coordinate system for each operation.

When you pick a coordinate system for an operation, the toolpath data is generated relative to it.



What: Create a volume clear operation.

How:



OpGroup Specification form

Name: Machine Cavity A

Operation Selection form

Category: *Milling*

Type: *Volume Clear*

Create

Don’t close the Operation Specification form.

What: Pick the coordinate system for the operation.

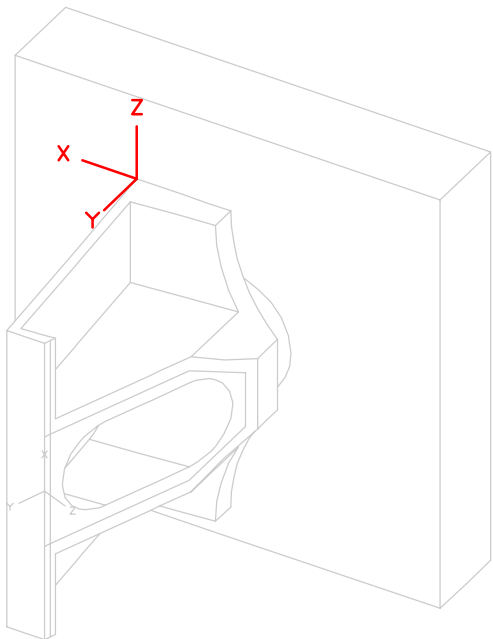
How:

Operation Specification form

Coordinate System List form

MK2

Dismiss



Things to notice

When you select MK2, the coordinate system is highlighted on the screen.

For volume clear operations, pick the coordinate system before selecting the surfaces. When a setup doesn't contain a stock, you must define one by specifying its shape and size. The software calculates the size of the stock in relationship to the current MCS. If you didn't pick the proper MCS before defining the stock, you may receive an error indicating that your values are reversed or incorrect.

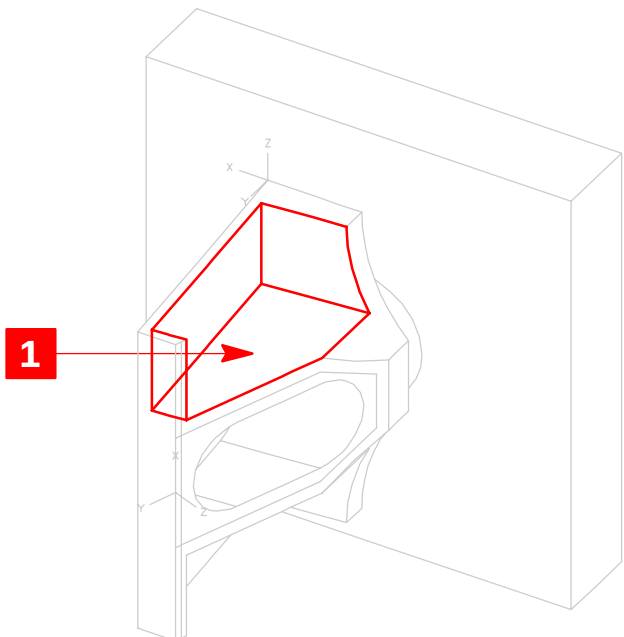
What: Name the operation and select the surfaces to be machined.

How:

Operation Specification form

Name: Volume Clear Cavity A

1 *F13* (double−click to select the cavity)



Don’t close the Stock Specification form.

What: Define the stock.

How:

Stock Specification form

Stock Top

1 *F16*

accept if necessary

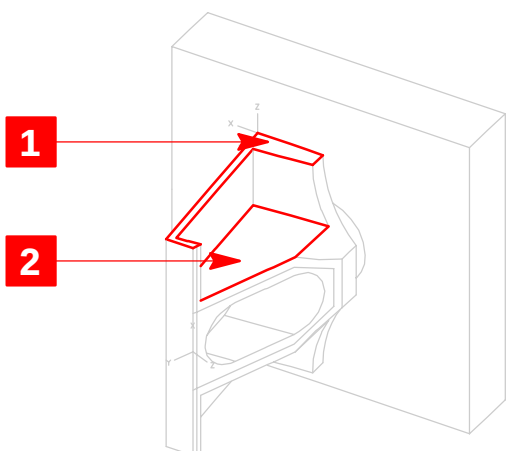
Stock Bottom

2 *F13*

accept if necessary

With XY Offset of: .25

OK



Don’t close the Operation Specification form.

What: Use a 1/2” diameter end mill to machine the cavity.

How:

Operation Specification form

Cutting Tool Specification—Mill form

Find...

Item Selection form

1/2” dia end mill

OK

OK

Don’t close the Operation Specification form.

What: Specify the cut pattern as *Spiral In*. Then set the cutting passes as .25” beyond the boundary of the stock.

How:

Operation Specification form

Machining Parameters form

Cut Pattern

Axial Depths...

Maximum Depth Of Cut: .5

OK

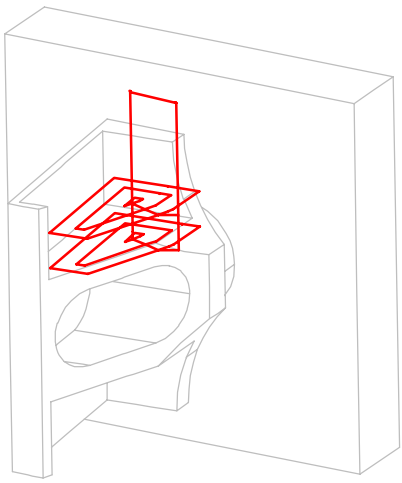
OK

Don’t close the Operation Specification form.

What: Generate the toolpath.

How:

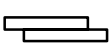
Operation Specification form



Things to notice

The engage and retract motions are parallel to the machining coordinate system that you selected for the operation. The toolpath also forms a spiral-in pattern and its cut passes extend beyond the edge of your defined stock.

Recovery Point



File
Save

What: Create a new opgroup and operation by copying the current opgroup. When you copy the operation, its tool and machining parameters are copied also. You'll only have to pick different surfaces and a new coordinate system to create a toolpath.

How:

NC Job Planning form

Machine Cavity A

Deselect *Machine Cavity A* by pressing the Control key and selecting *Machine Cavity A*.

Don't close the NC Job Planning form.

What: Modify OpGroup−2, then modify Operation−2.

How:

NC Job Planning form

OpGroup Specification form

Name: Machine Cavity B

Operation−2

Operation Specification form

Name: Volume Clear Cavity B

Don’t close the Operation Specification form.

What: Pick the coordinate system for the operation.

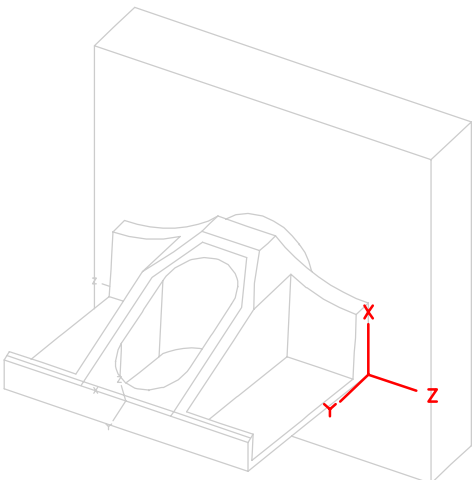
How:

Operation Specification form

Coordinate System List form

MK3

Dismiss



Don’t close the Operation Specification form.

What: Pick the surfaces to be machined.

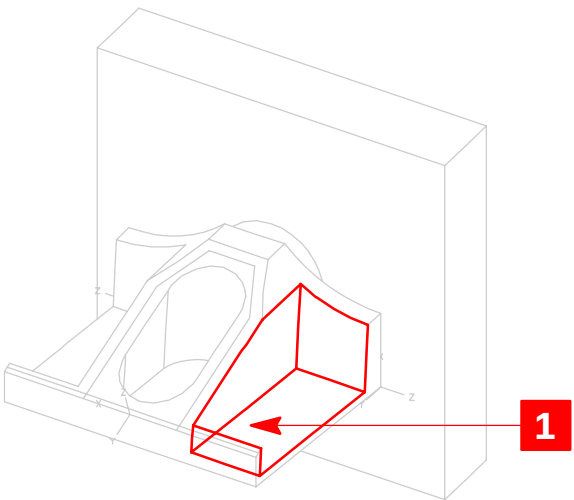
How:

Operation Specification form

Stock Specification form

Deselect All

1 *F4* (double–click to select the cavity)



Don’t close the Stock Specification form.

What: Define the stock.

How:

Stock Specification form

Stock Top

Deselect All

1 *F18*

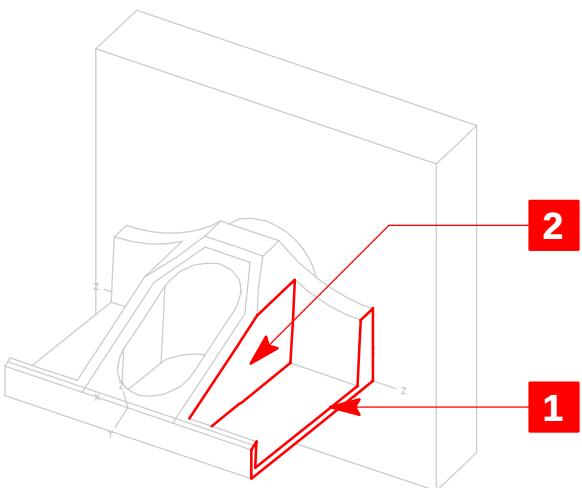
Stock Bottom

Deselect All

2 *F10*

OK

Don’t close the Operation Specification form.



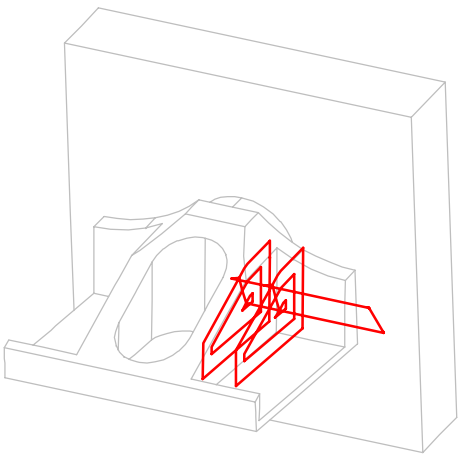
What: Generate the toolpath.

How:

Operation Specification form

I-DEAS Warning

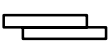
OK



Things to notice

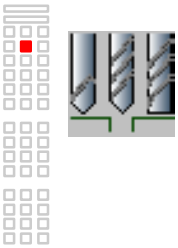
Because you copied the tool and machining parameters from the previous operation, this toolpath appears similar.

Recovery Point

 *File*
Save

What: Create an opgroup, then a manual operation.

How:



OpGroup Specification form

Name: Machine Face C

Operation Selection form

Category: *Milling*

Type: *Manual*

Create

Don’t close the Operation Specification form.

What: Pick the coordinate system for the operation.

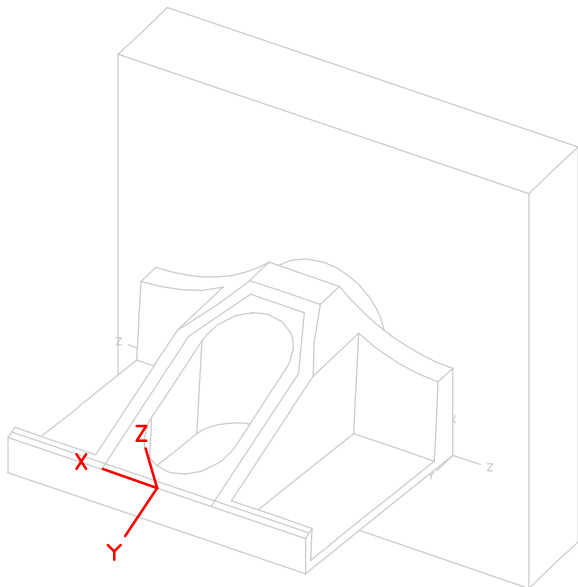
How:

Operation Specification form

Coordinate System List form

MK4

Dismiss



Don’t close the Operation Specification form.

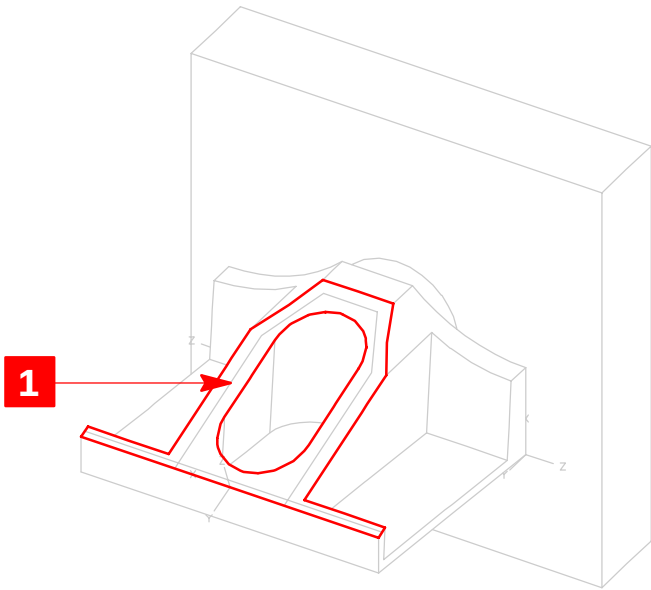
What: Pick the canted surface to be machined.

How:

Operation Specification form

Name: Manual Mill Face C

1 *F20*



Surface Selection form

Dismiss

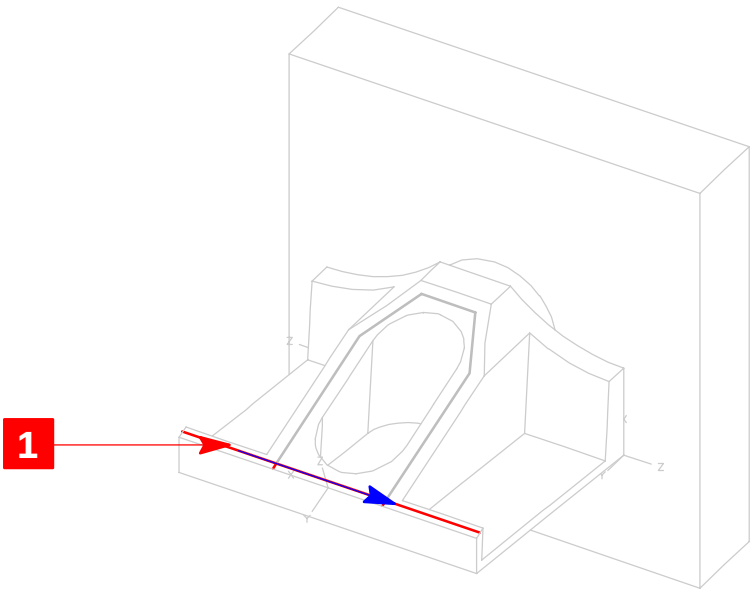
Don’t close the Operation Specification form.

What: Pick the first guide curve that defines the path of the tool. The arrow indicates the cutting direction of the tool, and the start point and entry of the toolpath.

How:

Operation Specification form

Machining Parameters form



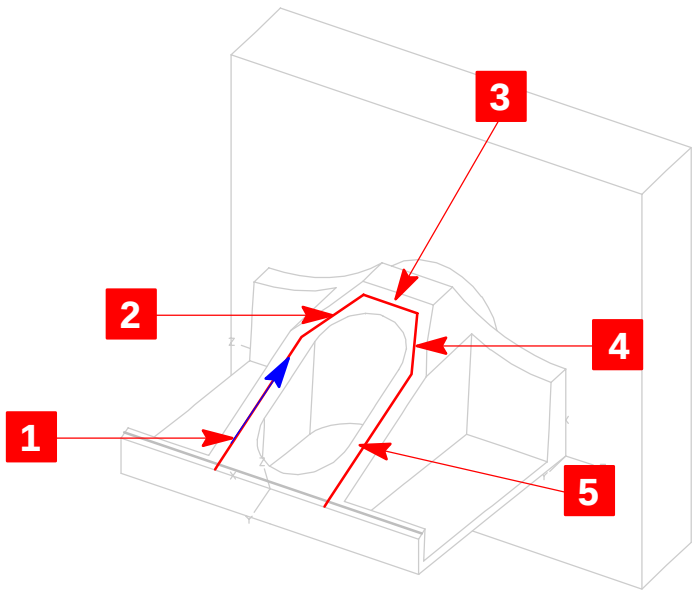
Guide Curve Sets form

Change arrow direction, if necessary.

Don’t close the Guide Curve Sets form.

What: Pick the second guide curve. The arrow indicates the cutting direction of the tool and the exit.

Guide Curve Sets form



Guide Curve Sets form

Change arrow direction, if necessary.

OK

Don’t close the Machining Parameters form.

What: Define an axial depth of zero so the tool contacts the surface of the part.

How:

Machining Parameters form

- *Project Onto*—Off
- *Single Pass with Z Translation of*: 0

Don’t close the Machining Parameters form.

What: Specify an in-plane entry and an in-plane exit. Also set the maximum drive length for region connections to 100 percent of the tool diameter so that the tool retracts between cuts.

How:

Cut...
Entry...

Entry Type: In Plane

Angle: 0

Entry...
Exit...

Exit Type: In Plane

Angle: 0

OK

Don’t close the Operation Specification form.

What: Use a 1” diameter end mill to machine the surface.

How:

Operation Specification form

Cutting Tool Specification—Mill form

Find

Item Selection form

1” dia end mill

OK

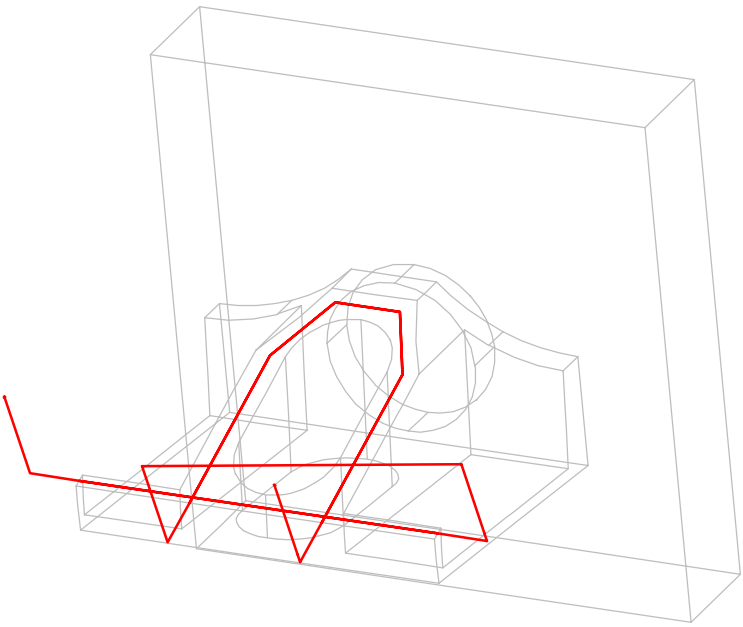
OK

Don’t close the Operation Specification form.

What: Generate the toolpath.

How:

Operation Specification form



Things to notice

The entry and exit of the toolpath follow the direction of the arrows on the guide curves that you selected earlier.

Recovery Point

File
Save

Tutorial wrap-up

You’ve completed the Indexing tutorial.